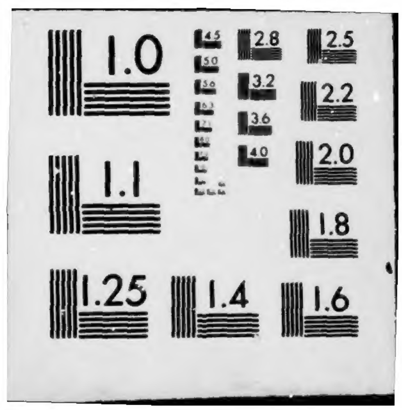
AD-A033 638   OFFICE OF NAVAL RESEARCH LONDON (ENGLAND)   F/G 20/4
EUROPEAN DEVELOPMENTS IN COMPUTATIONAL FLUID DYNAMICS,(U)
NOV 76 R H NUNN
UNCLASSIFIED   ONRL-R-10-76   NL

1 OF 1
AD-A
033 638

END
DATE
FILMED
1-28-77
NTIS


1.0
4.5
5.0
5.6
6.3
2.8
3.2
3.6
4.0
2.5
2.2
2.0
1.1
1.8
1.25
1.4
1.6

U.S. DEPARTMENT OF COMMERCE
National Technical Information Service

AD-A033 638

EUROPEAN DEVELOPMENTS IN COMPUTATIONAL FLUID DYNAMICS

OFFICE OF NAVAL RESEARCH,
LONDON (ENGLAND)

12 NOVEMBER 1976

ONR LONDON REPORT

ONRL R-10-76

OFFICE OF NAVAL RESEARCH

BRANCH OFFICE LONDON ENGLAND

EUROPEAN DEVELOPMENTS IN COMPUTATIONAL FLUID DYNAMICS

DR. ROBERT H. NUNN

12 NOVEMBER 1976

UNITED STATES OF AMERICA

This document is issued primarily for the information of U.S. Government scientific personnel and contractors. It is not considered part of the scientific literature and should not be cited as such.

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER ONRL R-10-1976	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) EUROPEAN DEVELOPMENTS IN COMPUTATIONAL FLUID DYNAMICS		5. TYPE OF REPORT & PERIOD COVERED 6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) ROBERT H. NUNN		8. CONTRACT OR GRANT NUMBER(s)
9. PERFORMING ORGANIZATION NAME AND ADDRESS Office of Naval Research Branch Office London, Box 39, FPO New York 09510		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
11. CONTROLLING OFFICE NAME AND ADDRESS		12. REPORT DATE 12 Nov 1976 13. NUMBER OF PAGES 28
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) UNCLASSIFIED 15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

APPROVED FOR PUBLIC RELEASE: DISTRIBUTION UNLIMITED

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Computational Fluid Dynamics
Numerical Methods
Transonic Flow
Viscous Flow

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

In Europe, as in the US, the use of the computer to solve complex problems in fluid dynamics is a burgeoning field of endeavor. This report briefly describes some of the current European efforts in computational fluid dynamics. The citations are intended as "leads" to the individuals and institutions involved in such activities, and recent references are given to provide guidance to the latest published information.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

ONRL R-10-76

EUROPEAN DEVELOPMENTS IN COMPUTATIONAL FLUID DYNAMICS

INTRODUCTION

The US visitor to Europe is likely to be impressed by a lot of things, particularly those ideas and institutions that stem from ancient origins or otherwise provide a contrast with the American way of life. In science and technology however, there are many areas of endeavor in which it would be a great mistake to think that European efforts lack the spirit of creativity and ingenuity that we like to associate with "good old Yankee know-how." A case in point is that of Computational Fluid Dynamics (CFD). In my visits to research centers and attendance at international symposia it has been easy to see that CFD and, in general, the exploitation of the latest computational capabilities, is a burgeoning activity in Europe. Even though my own background is not especially related to "C" part of CFD and I was therefore not particularly interested in seeking out research activities in this area, I was confronted at every turn with instances in which the computer was being put to good use in the solution of complex engineering problems. For this reason, and in spite of the aforementioned lack of expertise, it seems to me that a useful purpose might be served if these activities are recorded in a more-or-less systematic way.

The first problem has been with the goal of being "systematic." How should the various works be listed? By numerical approach, method of solution, nature of the problem tackled, country, source of reference...? The reader will see that the organization of the report is somewhat hybrid in this regard. Where a relatively large body of information was available (transonic flow, viscous flows), work in various countries has been listed under the problem category. However, the diversity of activity in CFD and the necessarily limited coverage thereof have often forced a departure from this system. The reader is cautioned to avoid the impression that the relative number of citations for a given country, problem area, etc., is a valid indication of level of activity. This is merely a reflection of the exposure that the writer has received during the last year. And this is probably as good a place as any to offer apologies to those workers whose important efforts have not been listed here. The report is simply the result of an effort to create a conglomeration out of the bits and pieces of available information.

The purposes of the report are to provide information by which CFD practioners can estimate the state-of-the-art in Europe and to furnish appropriate points of contact. It is sincerely hoped that contacts will be made, for it is evident that such interactions are highly valued by European researchers, and their US counterparts can count on a cordial and responsive exchange of information. Further, it should be pointed out that the brief descriptive paragraphs given here are by no means comprehensive (it is hoped that they are at least accurate), and valid comparative judgements can be formed only by means of direct interaction between parties concerned. Where possible I have tried to point out the unique and noteworthy features of the works cited. Even so, this report has developed into something of a "shopping list," a term that is perhaps regret-able, but nonetheless accurate. Finally, a remark should be made concerning the references. Much of the work in CFD in Europe stems from the extensive US developments in this field. A minimum number of US references is given, however, since again the purpose of the references is to provide a starting point for US researchers who want to learn more about developments abroad.

TRANSONIC FLOW

Interest in numerical treatment of the equations describing transonic flows was greatly stimulated in 1970 by the suggestion of Murman and Cole [1] that the discrete approximations be handled in a way that reflects the directional bias of the physical situation: central differencing in the subsonic regions where conditions at all neighboring points influence those at the calculation point, and the equations are of the elliptic type; and upwind differencing in supersonic regions where zones of influence are upstream of the calculation point, and the equations are hyperbolic. Since that time, in Europe and elsewhere, a good deal of effort has gone into the refinement of calculation schemes for these flows.

Several areas exist in which contrasts can be found between the approaches of various researchers. These include the governing equations themselves which are in general the potential flow relationships that neglect viscous and other dissipative effects, but, beyond this, can be any one of several simplified versions in which transonic small perturbation (TSP) methods have been used to eliminate all but the dominant terms. The assumptions governing the TSP scheme result in a variety of forms of the final result including the classical Guderley-von Karman equation, that used by Albone, Hall and Joyce at the Royal Aircraft Establishment (RAE) Farnborough, the version recently developed by van der Vooren and his coworkers at the National Aerospace Laboratory (NRL) in Amsterdam (discussed below), and several forms used in US studies.

In addition to the form of the equation being integrated, other considerations are the dimension of the flow (to my knowledge, the full potential flow equations have not been solved for 3-dimensional transonic flows, mostly because it is not a computationally practical thing to do); the method of switching between subsonic and supersonic regions (this is often handled by a numerical viscosity term that is injected into the finite difference equations for the calculation of supersonic points); the treatment of shocks (which must be weak in order to justify the potential flow model, but are necessary to accomplish the deceleration of supersonic flows); the handling of coordinate systems (such as the rotation of coordinates into alignment with the main flow direction so that the Murman-Cole type-dependent differencing can be employed in the integration of the full potential flow equations); and a variety of methods employed to solve the set of nonlinear partial difference equations by a convergent iterative scheme. Needless to say, with all these and more variations possible between solution methods

there are quite a few numerical models for transonic flows, and it is difficult to draw comparisons between them. Here the goal is to reveal some of the European approaches in the hope that comparisons will eventually be made through increased interaction between the involved parties.

Many fluid dynamicists have come to think of CFD as including only those techniques that are based upon finite differencing of the governing equations. In order to include a more comprehensive description of work underway in Europe, however, other methods will be noted here as well. These include, of course, the finite element method (FEM), the use of the hodograph approach and the method of integral equations. All of these ultimately lead to solution by computer, but they involve different treatment of the governing relationships prior to their submittal to the digital grinder.

The Federal Republic of Germany

At Dornier GmbH, in Friedrichshafen, W. Schmidt and R. Vanino have been applying the TSP equation to the numerical prediction of flows past various 3-dimensional flight configurations. Type-dependent differencing is utilized at mesh points of variable density within a rectangular region. Several different methods have been employed to implement the body boundary condition including the local use of higher order approximations for closed bodies. The line relaxation method gives stable convergence with over-relaxation in the subsonic region and under-relaxation in the supersonic part. The computer code is used in a design mode for the calculation of supercritical wing and wing-body configurations, as well as a basic tool for the study of transonic flow behavior. Several swept-wing and wing-body combinations have been submitted to computer prediction, and good agreement with experiment (where available) is obtained after the application of wind-tunnel wall corrections and adjustments for boundary layer displacement effects. These latter effects often have been observed to be important, and the Dornier group has begun to develop 3-dimensional integral methods for the application of appropriate boundary layer corrections. In addition, and in conjunction with N. Agrell of FFA in Sweden, work has been started on the use of a non-orthogonal coordinate system for aligning grid lines with swept leading edges. Although this is also done at NLR and RAE, the Dornier approach is to transform the difference equations rather than the basic governing equations [2,3,4].

The TSP equations are solved by integral methods by H. Hansen at the DFVLR Institute für Aerodynamik, Braunschweig. Standard procedures, employing Green's theorem, lead to the integral formulation for the 2-dimensional case. The method is used in the design mode by means of a sequence of reductions of the airfoil pressure distributions from the prescribed non-linear compressible form, through a linear compressible distribution, to a linear incompressible condition (the latter transformation is accomplished by the application of Goethert's rule). Comparisons with exact methods show acceptable agreement for both subcritical and supercritical airfoils, with and without lift [2]. Integral methods are also employed by A. Frohn, at the DFVLR Institut für Strömungsmechanik in Göttingen. She is currently working to refine the method of approximating the integral terms in such a way that the shock location is properly determined for transonic flow past a parabolic airfoil contour [2].

In the area of unsteady transonic flows, G. Redeker, also of DFVLR, Braunschweig, has applied boundary layer growth and separation models for the prediction of buffet onset on supercritical airfoil shapes. It is assumed that boundary layer separation is induced by the strong adverse pressure gradient existing on downstream surfaces, rather than by interaction with the imbedded shock (which is weak). Airfoil pressure distributions are computed by the method of Garabedian and Korn [5], and comparison with experiments show remarkably good agreement for the prediction of Mach number and angle of attack at buffet onset (which is defined by the forward movement of the separation to the 90% chord point) [2,6].

At the Institut für Mechanik, Technische Universität Hannover, I. Teipel has begun to develop a method for the prediction of transonic aerodynamic flutter by considering solutions to the time-dependent version of the TSP equations. "Local linearization methods" are used in both the subsonic and supersonic regimes. With freestream Mach number and thickness ratio as parameters, stability ("neutral damping") boundaries are determined for various hinge points [2,7].

R. Bohning and J. Zierep (Lehrstuhl für Strömungslehre U. Karlsruhe) use a 3-layer model to calculate the 2-dimensional flow associated with the interaction of a weak transonic normal shock with the turbulent boundary layer along a curved wall. Their results show considerable upstream influence along the contour surface and the remarkable, if by now well-known, "post-expansion" phenomenon in which the flow near the outer edge of the boundary layer is accelerated in regions just down stream of the shock [2,8].

At DFVLR, Göttingen, H. Sobiezcky solves problems in plane, compressible flow by means of a modified hodograph plane. The linearization effect of the hodograph method leads to two linear systems, one elliptic and the other hyperbolic, the solutions to which are coupled along the sonic line. The hyperbolic region is solved by a numerical characteristics method, and in the elliptic regions Sobiezcky suggests the use of the rheoelectric analogy for solution of these equations [2,9].

A. Eberle at Messerschmitt-Bölkow-Blohm (MBB) in Ottobrunn, has applied a further transformation to Sobiezcky's "rheograph" plane such that the geometry takes on that of a uniform flow past an appropriately deformed cylinder Converting to the integral formulation, Eberle uses a panel method (distributed singularities) to discretize the integrals and obtains solutions for the elliptic portion of the flow past an airfoil contour, including the sonic line, and the imbedded supersonic region is solved by the method of characteristics [2,10].

France

D. Euvrard (Ecole Nationale Supérieure de Techniques Avancés--ENSTA, Paris) and Y. Morchoisne (Société Nationale Industrielle Aérospatiale, Suresnes) calculate the symmetrical transonic flow past airfoils by a method that tests the iterative results in the elliptic region against asymptotic expansions. Subsequent iterations are based upon the minimization of the distance separating the two sets of data. Calculation downstream of the limiting characteristic is managed by means of the method of characteristics. The calculations are limited in application because shocks are not predicted except in a most rudimentary way. Even so, the scheme has been extended to airfoils at angle of attack by F. Grosjean and G. Tournemine at Brest University. Although the methods should be valid for the flow near airfoil leading edges, comparison with experimental data (NACA 0012 airfoil) shows only limited agreement with the stagnation point location and the pressure distribution over the leading 20% chord regions [2,11,12].

At ONERA, Chatillon, J. J. Challot has proposed a numerical treatment of the TSP equations that emphasizes fully conservative formulations, i.e., ones in which the finite difference formulation appears as an expression of conservation of dependent variables across mesh boundaries. Transonic flow in a constricted duct is calculated to demonstrate the efficacy of the method [2].

T. S. Luu and G. Coulmy (Laboratoire d'Informatique pour la Mécanique et les Sciences de l'Ingénier--LIMSI, ESN 30-8:361) have solved the full potential equation for flow in the transonic range through 2-dimensional cascades. The method employed appears to be equivalent to the most sophisticated techniques used in the US in that type-dependent differencing is used in a rotated coordinate system with a controlled numerical viscosity leading to automatic occurrence of shocks. The computational region is established by means of the solution of the equivalent incompressible flow problem expressed in stream function and velocity potential coordinates so that a "natural" coordinate system is developed in which the boundary conditions are exactly specified [2,13].

The finite element method (FEM) appears to be much in favor in France and has been used extensively in the analysis of transonic flows. R. Glowinski (Université de Paris), J. Periaux (Avion Marcel Dassault, Suresnes), and O. Pironneau (Laboria-Iria Le Chesnay) have approached the inviscid transonic flow problem from the point of view of optimal control theory in which the "cost" is the difference between the exact and the approximate solution. The resulting minimization criteria are discretized and satisfied by the FEM method. Although the work appears to be in its formative stages, the authors claim convergence in the mixed-flow regions, in the presence of shocks, for 2- or 3-dimensional flows past lifting bodies [14,15]. A more general discussion of FEM methods in use at Dassault, including the analysis of separated flows is given in [3] by P. Perrier and J. Periaux.

The Netherlands - The National Aerospace Laboratory (NLR), Amsterdam

The NLR is actively involved in transonic flow calculations utilizing both hodograph and finite difference techniques. In the former instances, J. W. Boerstoel, G. H. Huizing [2,16], and others at NLR have been active since the non-computer days when the hodograph approach was a necessity. Boerstoel is strongly of the opinion that such techniques are still useful, particularly in the design of shock-free airfoils (hodograph methods are limited to 2-dimensional shock-free flows). At NLR the design scheme begins with the guessing of an airfoil image in the hodograph plane. The boundary value problem is then solved, followed by mapping to the physical plane and comparison with desired airfoil shapes or pressure distributions. Iteration

proceeds until the desired design characteristics are obtained. An especially interesting capability exists at NLR in a hybrid procedure that begins with an airfoil design according to the hodograph procedure but is followed by a calculation of off-design behavior using finite difference numerical predictions coupled with boundary layer calculations and, where available, empirical inputs. This procedure is iterated until a suitable overall performance is predicted and, if necessary, verified in the NLR wind tunnels.

In the finite difference approach at NLR, J. van der Vooren and others [2] have recently adopted a new fully-conservative form of the TSP equation that is derived directly from the series expansion of the perturbed mass continuity equation. They have shown that this form is reliable in the calculation of shock discontinuities especially for a wide range of freestream Mach numbers and for shocks with appreciable sweep--situations in which one or all of the other forms are inadequate. This work is soon to be reported in the NLR technical literature.

Sweden

At SAAB-Scania AB, in Lincöping, B. G. Arlinger has extended his methods used for the transonic flows in and around inlets [17] to include the case of a supersonic freestream [2]. Following a transformation to the hodograph plane, finite difference solutions are obtained through line relaxation using rotated type-dependent differencing, and numerical viscosity for switching between subsonic and supersonic regions. Various equation formulations have been investigated, and it has been found that the calculation times decrease for less conservative forms and that calculated shock positions and strengths move upstream and become weaker--a result that is in agreement with those of other investigators for these inviscid calculations. Arlinger's results are in very good agreement with available experimental results.

Also at SAAB-Scania, Y.C-J. Sedin and K.R. Karlsson are working with a new approach to the axisymmetric form of the TSP equations [18]. In their method the equations are separated into two parabolic forms one of which is integrated inward from the far-field boundary while the integration of the other proceeds in an outward direction. Type-dependent differencing is used as well as sonic and shock-point operators based upon numerical viscosity. Consistent results have been obtained

in comparison with other methods and the rate of convergence is competitive. The method is currently being extended to 3-dimensional flow problems and initial results are promising [2].

The United Kingdom

At the RAE Farnborough there has been sustained interest in transonic flows on the part of several investigators. The latest version [19] of the solution of the 2-dimensional TSP equation is similar to the more or less "standard" methods [1] with the exception of a 9-parameter coordinate transformation that allows the solution to be pursued within a finite rectangular grid with grid-point density that can be varied to match the gradients in physical quantities. Agreement with more accurate methods [5] is essentially complete for relatively thick airfoils at small angles of attack with critical freestream Mach numbers. For the 2-dimensional viscous case, several methods have been used with success in order to correct the inviscid results for boundary layer displacement effects [20].

The main motivation at RAE in pursuing the 2-dimensional transonic problem is to formulate models that are accurate yet simple enough for extension to the real-world of 3-dimensional flows. The 3-dimensional work began in 1972, and much of it is reported in [21] for flow past wings, and by Albone, Hall, and Joyce in [2] for the flow past wing-body combinations. A modified second-order form of the 3-dimensional TSP equation is applied to account for the effects of wing-sweep. A crucial aspect of the RAE approach is, as in the 2-dimensional case, the creation of a coordinate system that combines efficiency with simplicity. In the 3-dimensional case this involves a coordinate stretching that leads to coordinates that conform to the leading and trailing edge contours of the wing. In validating the model the agreement with experiment is exceptionally good. In fact, Lock [2] states that it is "perhaps too good, considering that no allowance has been made for viscous effects." According to Lock, the list of priorities in future transonic flow should read: (1) 3-dimensional viscous effects, (2) application of the full potential flow equations with exact boundary conditions to 3-dimensional geometries, (3) improvement in the treatment of shock waves, including finite entropy and vorticity production, and (4) refinement of the prediction of turbulent boundary layer behavior including the regions of shock wave interaction and the trailing edge.

T. J. Baker (Aircraft Research Association Ltd., Bedford) has applied finite difference methods to the transonic flows past ducted bodies. The adoption of an entirely general (non-orthogonal) coordinate scheme allows flexibility and is more readily extended to deal with more complicated body geometries. Apart from this, Baker uses the TSP equations, and his approach to differencing and iteration is similar to other transonic techniques. Only the axisymmetric case has been considered [2], and with the exception of the usual difficulties regarding shock location, good agreement is obtained with experiments for subsonic approach Mach numbers. Other transonic flow analyses with particular emphasis on the treatment of coordinate systems are those of G. J. Clapworthy, P. W. Duck. and K. W. Margler at the University of Southampton. Here they have utilized elliptical coordinates to calculate the 3-dimensional flow past ellipsoids [2].

Finally, mention is made of the work continuing at Queen Mary College, London, where D. Nixon and G. J. Hancock have been developing extended versions of the integral method. This method is fully conservative and exceptionally fast, in comparison with finite difference methods; the latter improvement being largely due to the ability to calculate surface conditions without calculating the entire flow field. The latest extension to the theory [2] has been developed in order to obtain a more adequate representation of transonic flow with shocks. Improvement is indeed obtained, although the integral method tends to predict shock positions about 10% downstream from those predicted by the non-conservative finite difference schemes. This difference may not be important, however, since the use of the TSP equations for flows with shocks will inevitably be abandoned as more sophisticated shock treatment methods are developed. On the other hand, it is clear that for cases in which the TSP equations are adequate to represent the physical situation (i.e., small flow deflections with negligible shock and viscous effects) this work, and that of Hansen and Frohn (*op. cit.*), shows a definite computational cost-effectiveness accruing to integral methods.

Others

S. Nocilla and G. Geymonat (Politechnico Torino) and B. Gabutti (Università Torino) have developed a direct hodograph procedure that has been used to calculate transonic flow past

circular arc profiles. The flow is symmetrical for-and-aft even for the supercritical case since only zero-incidence shock-free flows are calculated [2].

At the Norwegian Institute of Technology, Trondheim, H. Nørstrud has obtained results using a simple approximating function to represent the flow field conditions for use in the basic integral equation formulation. For shock-free flows this method [22] shows essential agreement with that of Nixon and Hancock *(op. cit.)*, and Nørstrud claims to be able to obtain adequate results for flows with shocks.

VISCOUS FLOWS

Transonic flow analyses have been largely devoid of detailed treatment of viscous effects. In this section the problems discussed usually excluded consideration of transonic behavior; here the emphasis is described by a phrase that only a few years ago would have startled fluid dynamicists--"integration of the Navier-Stokes equations." Simplifying assumptions are still prevalent, however, including such things as restricted dimensionality, incompressible flow, and, above all, phenomenological turbulence models. In several European laboratories the FEM has become popular for the analysis of viscous flows but, with this exception, the chief differences between the numerical methods are dictated by the nature of the stated problem and its boundary conditions.

Much of what follows is derived from the presentations at the 5th International Conference on Numerical Methods in Fluid Dynamics on 28 June - 2 July, 1976 (ESN 30-8:369). Since the proceedings [14] have yet to be published, the references given here are often somewhat tenuous.

Belgium

At the University of Mons, G. Vanderborck and J. K. Platten have developed a computer program to test the linear hydrodynamic stability of a wide variety of internal plane and axisymmetric flows. Local potential theory, together with the self-consistent numerical scheme is used to predict critical Reynolds numbers for arbitrary wave numbers [14].

Ch. Hirsch, Vrije University Brussels, has applied the FEM to the solution of the Navier-Stokes equations for the 3-dimensional through-flow in turbomachinery. The program is applied to the analysis of the NASA TASK-1 single-stage transonic compressor and the NGTE-106 6-stage compressor. Remarkable agreement is attained for the time-averaged radial distributions of flow angles and velocities at various axial flow stations [3,23].

The Federal Republic of Germany

At the Technische Hochschule Aachen, Nan-Suey Liu studies the viscous flow through ducts whose cross sections vary with both space and time. The interest in the problem is motivated by a desire to understand the flows that occur in cardiovascular and bronchial systems. Liu writes the governing equations in a generalized non-orthogonal coordinate system and, through the application of Gauss' divergence theorem to a volume element in this system, obtains a set of explicit finite difference equations. A feature of his method is that the transformation of coordinate points from the physical space to the calculation grid requires no additional computer storage capacity. The computational method has been applied to several problems with known solutions (including Poiseuille flow and pulsating flow in a long straight tube), and the results prove the validity of his approach. Liu is currently calculating the flow to be expected in an arterial bifurcation [14].

Buoyant jets are a subject of great interest to several researchers at the University of Karlsruhe (Sonderforschungsbereich 80), and numerical methods have perforce been applied to the problem. The geometry is 2-dimensional and the governing equations are of a boundary layer nature in which the turbulence terms are modeled after the "k-ε" (k being the turbulence kinetic energy and ε the dissipation rate thereof) approach suggested by Launder [29]. M. S. Hossain and W. Rodi have shown [25] that the buoyancy source terms in the turbulence quantities are such that turbulence is enhanced in vertical jets and damped in horizontal jets. Their numerical method is the marching-forward finite-difference procedure of Patankas and Spalding [26] and yields satisfactory results in comparison with the limited experimental evidence available for the turbulence quantities in such flows.

J. J. McGuirk and Rodi have applied finite difference methods to the analysis of 3-dimensional heated surface jets discharging

into stagnant water. The novelties in their approach occur in the simplification of the full elliptic equations of motion so they can be treated more economically as parabolic, and the solution of two extra differential equations for turbulence properties at each point in the field. (It should be mentioned that both McGuirk and Rodi have been associated with D. B. Spalding at Imperial College, London.) Comparisons with experiment yield good agreement for the rates of vertical and horizontal jet spread, but the rate of axial temperature decay is poorly predicted. The latter effect is judged to be due to the details of the turbulence modeling (a safe estimate, since 9 empirical constants are involved). The approach of McGuirk and Rodi certainly appears competitive with those of other investigators, especially when efficiency of computation is taken into account [25].

France

At ONERA, various finite difference schemes have been used by H. Viviand and W. Ghazzi for the solution of the Navier-Stokes (NS) equations for compressible flows at high Reynolds numbers. Their method is characterized by a time-dependent coordinate transformation that retains the Cartesian velocity terms (so that only derivatives of new coordinates with respect to old coordinates and time are involved) and a highly flexible and rapid technique for mesh refinement. The calculations were at first carried out using MacCormack's 2-step method [27] for the entire field, but it has been found that efficiency is gained by employing a 1-step semi-implicit scheme in the wall viscous layer. The program has been used to solve the supersonic flow past a blunt body (cylinder) at Reynolds numbers up to 4×10^4, and current efforts are concerned with axisymmetric flows such as those past a blunt body with a flare (and, possibly, separation) [25].

Fluid motion which is driven by mobile immersed particles is studied by R. Peyret (U. Pierre and Marie Curie, Paris) and S. Childress (Courant Institute, New York). Buoyancy-driven as well as randomly-driven particles are considered in the incompressible unsteady case with 2-dimensional geometry. The particles are handled in a manner similar to the MAC (marker and cell) method, and the NS equations are solved using an implicit scheme involving successive iterations at each time step. It has been found that the various evolving patterns taken by the particles can be described by the ratio of the Richardson

number to the Reynolds number (~buoyant/viscous effects) and that these patterns are similar to the motions observed in cultures of certain microorganisms under the influence of gravity [25].

A considerable amount of CFD also goes on at the Centre d'Etudes Aerodynamiques et Thermiques (CEAT) in Poitiers. Especially significant are the numerical experiments, carried out in the group headed by Tsen Li Fang, in which fast Fourier transform and over-relaxation methods are used to obtain appreciable reduction in the time required to solve viscous time-dependent problems. The method is being applied to the numerical simulation of plane turbulence in the particular case of the evolution of flow behind a bluff body (see also ESN 30-9:406).

The United Kingdom

The activities with the Department of Mechanical Engineering at the Imperial College, London, are among the most prominent in the field of CFD for the solution of engineering problems. Under the direction of D. B. Spalding there has been a steady series of developments, each improving on previous studies, over about the last 10 years. The work is generally concerned with turbulent transport phenomena and within this area are included almost all aspects of compressibility, dimensionality, steadiness, species, and phases. For this reason a separate report is planned as a follow-on to the report of D. F. Dyer (ONRL Report R-2-73). Only a few of the most recent projects will be described here.

Spalding and U. Svensson (University of Lund, Sweden) have cooperated in the creation of a new model for the development and erosion of thermoclines in lakes and oceans. The analysis utilizes exchange coefficients that are calculated from a turbulence model [28], and this is claimed to be the main source of virtue in the method. At its present state of refinement the model gives qualitatively satisfying results for the growth of a thermocline due to radiation and wind-shear inputs. Comparison with the limited available data shows some qualitative deficiences which are partially overcome by the adjustment of parameters [25].

The group at IC is also moving into the area of turbulent 2-phase flows with buoyancy, and in his recent survey [25], Spalding outlines the difficulties involved, methods available,

and required extensions to existing analysis. There are many "gray" areas (such as the modeling of the two-phase effects of body forces, surface tension, turbulence transport, and interphase slip), but Spalding is of the opinion that sufficient information is available to warrant the birth of computer models to handle these problems. In fact, such models have been conceived at IC and are presently being tested against available data; they are also expected to point the way towards meaningful extensions of physical evidence needed to describe the various phenomena involved.

At the Research Department of the Central Electricity Generating Board, Leatherhead, D.A.H. Jacobs employs numerical modeling to predict the downwind dispersion and deposition of gaseous pollutants from industrial sources. The steady convective diffusion equation is modeled for 2-dimensional flows in which the diffusivity is caused to vary in either or both directions. The model has been used mainly in a parameter variation mode to estimate the relative influence of such things as wind speed, rate of ground level deposition, rain removal (washout), eddy diffusivity, and source height. Some support for the accuracy of the calculations has been derived from comparison with other predictive models [25].

The USSR

B.L. Rozhdestvensky and his comrades at the Computing Center of the USSR Academy of Sciences in Moscow have developed an intriguing approach to the evaluation of finite difference schemes for use with the analysis of high Reynolds number flows. The finite difference equations to be tested are subjected to linear stability analysis, and the spectral functions resulting are compared with the responses of the appropriate linearized versions of the NS equations, especially those which grow in time. The requirement is that for a finite difference scheme to be acceptable, its own stability characteristics must match those of the basic governing equations. If the test is passed, the scheme is applied up to the critical Reynolds number. The group has used plane Poiseuille flow as a vehicle for testing the method. Their involvement with spectral functions has led them to consider the nature of turbulence, and they have proposed some simple models of turbulent flow [14].

At the Siberian Division of the USSR Academy of Sciences O. F. Vasiliev, *et al.* have continued to refine their predictions

of the growth of a "turbulated" region in a stratified fluid [29]. The principal feature of their most recent efforts [25] has been a modification of the normal Reynolds stresses to include a directional bias due to buoyancy effects. The governing equations are for 2-dimensional unsteady turbulent flow, without shear (the fluid is initially at rest), and the Boussinesq approximation is applied to account for density variations. The turbulence diffusion coefficients are modeled after Launder, *et al.* [30]. Straightforward but sophisticated discretization and iteration techniques are used to obtain stability and convergence of the solution. After an initial period in which buoyancy appears to have little effect, the turbulated region is seen to grow horizontally faster and vertically slower than it would in homogeneous fluid. Clear evidence is seen of internal density waves within the fluid as counterrotating vortices are formed and driven off horizontally within the disturbed region.

B. M. Berkovsky and V. K. Polevikov (Luikov Heat and Mass Transfer Institute, Minsk) have developed a quite general method for the construction of higher order difference schemes for use with multi-dimensional problems in convection. In application, they have considered the problem of vertical free convection at Rayleigh numbers of up to 10^{10}. Their scheme, which is second order accurate in time and fourth order accurate in space, is compared with lower order schemes, and they show that for Rayleigh numbers greater than about 10^5 the higher order of approximation is essential to obtain reasonable accuracy with acceptable storage requirements (grid spacing) [25].

Others

At the Helsinki University of Technology, M. Lindroos and S. Laine have numerically investigated the stability of a boundary layer in a 2-dimensional incompressible laminar flow past a flat plate with a bump. The NS equations are cast in the vorticity-stream function form and solved by a finite difference methods in a region surrounding the bump. The solution scheme involves the combination of a variety of well-known methods (Crank-Nicholson with predictor-corrector procedures) and provides results for Reynolds numbers (at the bump) of up to 6×10^5. Not surprisingly, they observe that a critical Reynolds number exists for each bump height. Above these values fluctuations are observed behind the bump that resemble Tollmien-Schlichting waves [14].

Three-dimensional turbulent buoyant flows, of the type that occur in heating and ventilating systems, are matters of

special interest to B. H. Hjertager and B. F. Magnussen at the Norwegian Institute of Technology, University of Trondheim. Again, the k-ε turbulence modeling and numerical techniques developed by the group at IC are used. Their numerical predictions are significant if only in the sense that they can even be performed with meaningful results. Only 9 grid points are used in each of 3 coordinate directions for room dimensions 2.4 x 2.9 x 5.6 m; in the long dimension the average distance between points is 0.7 m, but in fact, because of grid concentrations near the walls, some points are separated by much larger distances! Good qualitative results are obtained, however, especially for the purpose of determining the relative effects of buoyancy and inlet air conditions [25].

NUMERICAL METHODS

All of the activities previously referenced have in some way, and usually in depth, involved the application of numerical methods. The means, however, have been of secondary interest with respect to the ends: the solution of specified problems in fluid mechanics. In this section are listed a few of the European works that, while having in mind the field of flow, are mainly concerned with numerical procedures. Since these are relatively few in number, the country categorization is abandoned in favor of the two basic methods: finite difference and finite element. Except where otherwise indicated, the investigations mentioned here are documented in [14].

Finite Difference Methods

Yu. I. Shokin, at the Siberian Branch of the USSR Academy of Sciences, has conducted a study to evaluate the influence of numerical viscosity on the dissipation and dispersion generated within various finite difference formulations. His particular interest is in difference schemes in which the numerical viscosity has a polynomial form, and he has evaluated the effect of these assumptions upon gas dynamic behavior. Two-dimensional gas dynamic problems are also the vehicle for the methods of V. V. Rusanov (Inst. Applied Math., Moscow) which are third-order accurate [31]. He has considered stability problems, and, in particular, the influence of the numerical filtering of impulses and shock waves injected at various directions with respect to the mesh orientation [14].

In Germany, at the Institut für Reaktorentwicklung Karlsruhe, U. Schumann has adapted odd-even reduction methods and the capacitance matrix approach to formulate a code for directly solving the Poisson equation in 2-dimensions with Neumann, Dirichlet, or periodic boundary conditions [14].

A classical problem in CFD, that of scales of influence for the 2 basic governing equations (vorticity transport and the Poisson equation for stream function), is the subject of the investigation by F.P.H. van Beckum of Twente University of Technology, The Netherlands. To skirt the problem, van Beckum has been using different coordinate systems for the 2 equations. For the vorticity transport equation, coordinate lines are concentrated in regions of high influence, such as the wake. In each iteration the entire vorticity and stream function fields are calculated separately by a direct method. Numerical experiments have been performed to investigate the efficacy of the method when applied to the governing equations with a specified velocity field (thereby linearizing the vorticity transport equation).

At the University of Liège, in Belgium, J. J. Portier has conducted a carefully controlled numerical experiment to evaluate the relative merits of upwind and central differencing schemes in the calculation of vertical free convection processes. Although the "winner" in the comparison is not obvious, it is clear that the 2 methods converge to different solutions with the discrepancies becoming somewhat drastic at higher Grashof numbers (values up to 10^7 were tested). In fact, Nusselt-Grashof correlations resulting from the tests were

$$Nu = 0.163\ Gr^{0.287}, \text{ and}$$

$$Nu = 0.204\ Gr^{0.270}$$

for the central and upwind differencing formulas, respectively. The work is best considered as a warning to numericists that all is not always as it seems in the selection of differencing techniques--additional experimentation is needed. A particular limitation on the present work is the relatively coarse (25 x 25) uniform grid employed in both methods so that it is not clear if, in fact, either method is actually convergent [25].

Finite Element Methods

The method of J. Roux, Institute de Mècanique des Fluides, Marseille, involves the formulation of a variational inequality stemming from the Poisson equation. For problems in fluid mechanics, the latter is obtained through the application of the Chaplygin hodograph transformation to the equations of motion of compressible inviscid 2-dimensional flows. The solution of the inequality is computed numerically by means of an appropriate FEM, and the results have been tested, with success, against experiments with the NACA 0012 airfoil. The theory is valid up to critical freestream velocities.

At the Ruhr-Universität Bochum, in West Germany, G. Schmid is interested in the special advantages and shortcomings of the Ritz and Galerkin methods when applied to problems in fluid mechanics. He has compiled an extensive review of the 2 different approaches, and his general conclusions support the findings of others: the Eulerian description normally employed in flow problems leads to a preference for the Galerkin technique (which is not dependent on the existence of a general variational principle) in formulating the FEM to be used for solution.

The variational technique is, however, coupled to the E Eulerian description in the work of A. Ecer and H. Güvenen at the Middle East Technical University in Ankara, Turkey. A mixed Eulerian-Lagrangian approach is used, and considerable economy is realized by the elimination of pressure as a dependent variable and the reduction of the Lagrangian description of the velocity vector to a set of linear algebraic equations. The method appears to hold great promise for the analysis of viscous flows past complicated geometries such as lifting surfaces with separation [12,32].

OTHER TOPICS OF INTEREST

Inevitably, the categories given above are inadequate to describe the nature of some of the CFD projects underway in Europe. In this section an effort has been made to bring together those efforts that, however important, do not seem to be part of the relatively widespread European involvement in transonic flows and, in one way or another, the numerical integration of the NS equations.

Panel Methods

Panel methods have become somewhat classical in comparison with finite difference schemes: they are well-developed, reliable, and still the standard method for aerodynamic design except where extensions are needed to predict such special effects as transonic behavior and viscous interactions. The methods are all based upon potential-flow ground rules, and most consist of the distribution of singularities within and on body contours in such a way that boundary conditions are satisfied. The "panels" in the panel method represent the discretized regions of the body surface within which the singularities are located. The integral equations that result from the summation of the effects of the singularities are such that the panel method is closely allied to the method of integral equations.

In the subsequent paragraphs only a few European applications are mentioned, mainly because the references available for them are relatively recent. This should not be taken to represent anything like the total European activity in panel methods which, as noted above, are in common usage today.

At Messerschmitt-Bölkow-Blohm (MBB) in Ottobrunn, (Munich), W. Kraus and his coworkers [3] have developed the panel method to the point that it is generally used as a preliminary design tool for 3-dimensional aerodynamic bodies in subsonic, supersonic, and to some extent, transonic flows. Lifting bodies are treated (via the Kutta condition) by the inclusion of a distribution of doublets on thin sheets that extend from the trailing edge into the wake and upstream internal to the body contour.

For subsonic compressible flows, the MBB employs a modified version of the Göthert rule to correct the corresponding incompressible calculations. For supersonic flows, the most recent method [33] utilizes source-sink distributions on the body panels and vortex distribution on the wing contour. For transonic flows, the method of Eberle [10] is employed in which the panel method is applied to discretize the integral equations in the hodograph plane. The sonic line is determined and the method of characteristics is used to extend the calculations into the embedded supersonic region. The efforts at MBB are closely coordinated with those at the Dornier Company in Friedrichshafen and the DFVLR-Braunschweig. These have resulted in an extremely versatile capability, always limited by the potential flow assumption, but including interference effects, the calculations of the entire flow field (not just the body surface conditions), the behavior of external stores (before and after launch), and the influence of jet intakes and exhausts. Although there is research interest in such improvements

as the use of curved panel elements with varying singularity distributions (which allows the use of fewer panels), these have not yet found application in the MBB programs.

The panel method is used at the National Aerospace Laboratory (NLR) in Amsterdam by W. Loeve, J. W. Sloof, and others [34]. Here again the method has been widely applied to a variety of 3-dimensional geometries. Transonic finite difference methods are also well developed at NLR, and there is a tendency towards a hybrid approach to the calculation of the flow past bodies over a range of subsonic through supersonic Mach numbers. Loeve has recently conducted a comparative review of the state-of-the-art in panel methods and field calculation methods [25].

Supersonic Flows

Major problems of current interest in supersonic flows are associated with the occurrence of shocks and their interaction with and effect upon the flow along the surface of the body. The shock wave-boundary layer interaction is under study at several European institutions, a few of which have been mentioned in previous sections of this report. As far as the numerical treatment of shocks in inviscid supersonic flow is concerned, there are 2 recent developments in the UK. At the University of Salford F. Walkden and D. Evans have extended their shock-capturing methods [35] to allow the treatment of shocks as discontinuities in the finite difference solutions of the governing hyperbolic partial differential equations [14]. Their approach includes both shock detection and continuation of the flow calculation with the shock as a discontinuity, and is applied to 3-dimensional flows.

A new shock-capturing algorithm has been developed by P. L. Roe at the RAE in Bedford [14]. The algorithm is based upon forms of the difference equations which hold along characteristic lines so that the shock waves are exactly captured if they are aligned with the calculation mesh. When this is not the case, test problems have shown that the shock profiles are spread over about 3 mesh intervals and are propagated at the correct velocity. The extension to multi-dimensional problems is the current thrust of the study.

At the Politecnico di Torino, M. Pandolfi has constructed a numerical method that allows the calculation of 3-dimensional supersonic flows past elliptic cones with semiaxis ratios as

high as 10 (previous methods allowed values of up to about 4). According to Pandolfi [14] the success of his method stems from the use of a "proper" set of elliptic coordinates and selection of streamline slopes as the main dependent variables for the problem.

Miscellaneous

In previous paragraphs there has been occasional mention of some Russian contributions to the field of CFD. In fact, in the USSR there is a substantial capability for the numerical calculation of most of the "popular" flow situations: transonic flows, separated flows, turbulent flows, and various combination thereof. These capabilities, and the history behind their development, have recently been described by O. M. Belotserkovskii of the Computing Center, Academy of Sciences of the USSR, in Moscow [3].

A numerical technique that is currently finding widespread application in the USSR is the so-called "large particle" method that is a modification of the well-known "particle-in-cell" (PIC) method developed at Los Alamos. In the large particle approach [38] the particle is taken to consist of the entire mass of material occupying a computational cell at a given point in time. The evolution of the solution of the flow field is split into 2 parts at each time increment: the adjustment of the internal state (pressure, density, temperature), followed by a Lagrangian displacement of the particles based upon their instantaneous velocities and the chosen time increment. Recent discussions of the method, applied to problems in rarefied gas dynamics, are given by Belotserkovskii and, from the Siberian Branch of the USSR Academy of Sciences, Yanenko, *et al.*, in [14].

Also at the Moscow Computing Centre, V. P. Korobeinikov and his coworkers have numerically studied the propagation of shock waves in order to predict the effects of cosmic phenomena such as solar flares and the blast of a meteorite. The method of large particles is again applied and, in the case of the solar flare, is used to simulate the events that follow an abrupt release of energy in a finite volume near the sun. The influence of interplanetary magnetic fields are considered in these 2-dimensional calculations. The blast of a meteorite is simulated by an explosion of a semi-infinite cylindrical charge inclined to the Earth's surface and with an axial distribution of intensity.

The variable properties of the earth's atmosphere are considered with a view towards estimating the effect of the shock as it traverses the surface. A special interest of this work is to model the "Tunguska Catastrophe" which followed the impact, in 1908, of a meteorite in the permafrost region of Siberia; trees were felled over a radius of about 50 km, and the resulting worldwide seismic tremors led to an estimate for the mass of the meteorite of about 10 million tons [14].

Finally, mention is made of the work of M. Lenoir, of ENSTA, Paris, who has applied numerical computations to model the collapse of a cavitation bubble. The conditions treated include the presence of non-condensable gases within the bubble, and the proximity of solid walls. Beginning with a spherical bubble and appropriate pressure boundary conditions, the ensuing flow field is assumed to be irrotational, and the solution of the equations of motion at each instant of time is obtained by the finite element method which evolves through the formulation of a variational problem using distributed singularities. The method appears to be quite efficient and leads to results that show good agreement with experiment [14].

SUMMARY

The European effort in CFD appears to include a more-or-less full "bag of tricks." This is especially true from the numerical methods point of view. Thus researchers from several countries are fully conversant with such things as type-dependent differencing schemes(used in conjunction with rotated coordinate systems) and a variety of modern discretization and solution procedures. A particularly noteworthy indication of European maturity in CFD is the now common practice of using variable-density meshes and coordinate systems that are warped to facilitate application of the boundary conditions. As is common in CFD, much emphasis is placed upon computational cost effectiveness: time and storage vs accuracy. This emphasis is perhaps behind a continued high level of interest in more analytical and less numerically brutal schemes, such as those involving the hodograph and the methods of integral relations.

In the area of large scale calculations based largely upon finite difference methods, the lead of the US is apparently in no immediate jeopardy. But this is due in part to the relative size of the US and European computers and associated peripherals including professional staff. The ingenuity of several of the

European computing schemes recommends against complacency on the part of those who like to compare capabilities on a national basis.

Concerning the priority of problems yet to be solved, there appears to be fair agreement within Europe, and these priorities are not far out of line with those of the US CFD community. These are: (1) the treatment of viscous effects, including the need for improved turbulence modeling techniques and 3-dimensional boundary layer theories, (2) improved numerical techniques with special applications to 3-dimensional flows, and (3) improved criteria for the treatment of shock waves and their interaction with the boundary layer. A different kind of necessity is that for continued (and perhaps renewed) efforts to orient computer codes to the technically literate but numerically uninformed user. Finally, I found in Europe what I consider to be a healthy awareness of the need for more and better experimental data. The ever-increasing tendency to "validate" one computer code by comparing it with another is seen by many as an indication that the ability to compute has outstripped the ability to measure--to an extent that several fields of endeavor are weakly supported, if at all, by physical evidence.

REFERENCES

1. Murman, E. M. and Cole, J. D., "Computation of Plane Steady Transonic Flows," *AIAA J.* 9, 1, (1971), 114-121.

2. *Symposium Transonicum II*, ed. by K. Oswatitsch and D. Rues, Springer-Verlag, ISBN 0-387-07526-7, New York, 1976.

3. Notes for Lecture Series 87, *Computational Fluid Dynamics*, Von Karman Institute, Rhode St. Genese, Belgium, March 1976.

4. Schmidt, W., and Vanino, R., Dornier FB 74/43 B, Dec. 1974.

5. Garabedian, P. R. , and Korn, D. G., "Analysis of Transonic Airfoils," *Comm. Pure Appl. Math.* 24 (1971) 841-851.

6. Thomas, F., and Redeker, G., "A Method for Calculating the Transonic Buffet Boundary Including the Influence of Reynolds Number," *AGARD* Conf. Proc. 83 (1971).

7. Teipel, I., "Die Instationären Luftkräfte bei der Machzahl 1," *Zeitschrift für Flugwissenshaften* 12, Heft 1, (1964) 6-14.

8. Bohning, R., and Zierep, J., "Der Senkrechte Verdichtungsstoss an der gerkrümmten Wand unter Berücksichtigung der Riehbung," to appear in ZAMP.

9. Sobieczky, H., "Entwurf überkritischer Profile mit Hilfe der rheoelecktrischen Analogie," DLR-FB (1975), 25-43.

10. Eberle, A., "Eine exakte Hodographenmethode zum Entwarf überkritischer Profile," MBB VFE 1168-750, (1975).

11. Grosjean, F., "Construction analytique d'un é coulement sonique autour d'un profil symetrique, avee application an contrôle d'une méthode numérique directe," *Journal de Mécanique* 12, 3, (1973), 405-318.

12 Euvard, D., and Tournemine, G., "Méthode directe de calcul de l' écoulement sonique antour d'un profil d'aile donnée," *Journal de Mécanique* 12, 3, (1973), 419-61.

13. Luu, T. S., and Coulmy, G., "Calcul de l'ecoulemart transonnique avec choc à travers une grille d'aube," ATMA, 1975.

14. 5th International Conference on Numerical Methods in Fluid Dynamics, June-July, 1976. Preceedings to be published by Springer-Verlag.

15. Periaux, J., "3rd Analysis of compressible potential flows with the F.E.M., "*Int. J. Num. Meth.* 9, 4 (1975).

16. Boerstoel, J. W., and Huizing, G. H., "Transonic Shock-Free Aerofoil Design by an Analytic Hodograph Method," AIAA 7th Fl. Plasma Dyn. Conf., AIAA paper 74-539, (1974).

17. Arlinger, B. G., "Calculation of Transonic Flow Around Axisymmetric Inlets," AIAA paper 75-80, (1975).

18. Sedin, Y. C-U., "Axisymmetric Sonic Flow Computed by a Numerical Method Applied to Slender Bodies," *AIAA J*, April, (1975), 504-511.

19. Albone, C. M., Catherall, D., Hall, M. G., Joyce, G., An Improved Numerical Method for solving the Transonic Small Perturbation Equation for the Flow Past a Lifting Aerofoil, RAE Technical Report 74056 (1974).

20. Firmin, M.C.P., and Jones, A. F., The Calculation of Pressure Distributions, Lift, and Drag on Single Aerofolis at Supercritical Speeds, RAE Technical Report, (1975).

21. Hall, M. G., Firmin, M.C.P., "Recent Developments in Methods for Calculating Transonic Flow Over Wings," ICAS Paper 74, (1974).

22. Nørstrud, H., "The Transonic Aerofoil Problem with Embedded Shocks," *The Aero. Quart* 24, 2, (1973), 129-138.

23. Hirsch, Ch., and Warzee, G., "A Finite Element Method for the Axisymmetric Flow Computation in Turbomachines," *Int. J. Numerical Methods Eng.* (to be published). Also VUB Report STR-5, (1974).

24. Launder, B. E., "On the Effects of a Gravitational Field on the Turbulent Transfer of Heat and Momentum," *J. Fluid Mech.* 67. (1975), 569-581.

25. *Turbulent Buoyant Convection*, ICHMT International Seminar, 30 Aug-4 Sep, 1976, Hemisphere Publishing Corp., Washington, D.C.

26. Pantakar, S. V., and Spalding, D. B., *Heat and Mass Transfer in Boundary Layers*, Intertext, 2nd Ed. (1970), London.

27. MacCormack, R. W., "Numerical Solution of the Interaction of a Shock Wave with a Laminar Boundary Layer," in *Lecture Notes in Physics* 8, M. Holt ed., Springer-Verlag, New York (1971).

28. Launder, B. E., and Spalding, D. B., *Mathematical Models of Turbulence*, Academic Press, New York (1972).

29. Vasilief, O. F., Kuznetsov, B. G., Lythin, Y. M., and Chernykh, G. G., "Development of the Turbalized Region in Stratified Medium," *Proc. Int'l. Sym. on Stratified Flows*, ASCE, New York (1973).

30. Launder, B. E., Reece, G. J., Rodi, W., "Progress in the Development of a Reynolds-stress Turbulence Closure," *J. Fluid Mech.* 68, 3 (1975), 537-566.

31. Rusanov, V. V., "Non-Linear Analysis of the Shock Profile in Difference Schemes," in *Lecture Notes in Physics* 8, M. Holt ed., Springer-Verlag, New York (1971).

32. Bratanow, T., and Ecen A., "On the Application of the Finite Element Method in Unsteady Aerodynamics," *AIAA J*, 12, 4 (1974), 503-510.

33. Kraus, W., "Ein allgemerines Panelverfahren zur Bereckmung der dreidimesionalen Potentialströmung um beliebige Flügel-Rumpt-Leitwerkskombinationer um Unter-und Überschall (mit Programnbeschreibung)," MBB-VFE-1136 (1974).

34. Loeve, W., and Sloof, J. W., "On the use of Panel Methods for Predicting Subsonic Flow About Airfoils and Aircraft Configurations," NLR-MP-71018-U (1971).

10

35. Walkden, F., Caine, P., and Laws, G. T., "A Locally Two-Dimensional Shock Capturing Method for Calculating Supersonic Flow Fields," U. Salford F.M.C.C. Tech. Rep. No. 16176 (1976).

36. Belotzerkovskii, O. M., and Yanitsky, V. E., "Statistical 'particle-in-cell' method for the solution of the problem of rarefied gas dynamics," *Zh. Vych. Mat. i Mat. Fiz.* 15, 5, 1195-1208 (part I), and 6, 1553-1567 (part II), (1975).